In this section I'm going to show you how to restart HYSPLIT from a PARDUMP file. This section requires that you have completed the last one. That is you have the particle simulation, the volcanic simulation after 12 hours so that you have an output frame that is valid at 12 UTC on April 14th. And that we had plotted that graphic showing the particles distributed from the ground to 6000 m.

Now the PARDUMP file that was created is not just a file of particle positions, but it contains all the information that you would need to restart a HYSPLIT calculation. It contains the mass on each particle, its age, its height, density, if it has a distribution, the size of the distribution, and so on. All that information is contained in the PARDUMP file.

We're going to use that to restart the model with all the particle positions that existed at 12 UTC, but this time we're going to change the release height. So instead of releasing from 6, or up to 6,000 meters, we're going to release a 12,000 meters. And what we're doing is manually constructing a complex emission scenario. Now we could change other things besides the release height, obviously. We can, you can, change the release rate, we can change even the release location, if you have a moving source. But to do this calculation, so what we're going to do is go to the set up run menu, and instead of starting at zero, we're now going to start at 12. And instead of the starting location going to 6 km, let's say it goes to 12. We will still run for 12 hours, and we will be

using the same meteorology. So everything else stays the same. We still have those four particle size distributions. And since we didn't identify time in the start and stop, so this to be valid for this run as well, these fields are zero, which means they apply for the model start time.

And if you recall in the name list, on menu number nine, we output the PARDUMP file after twelve hours, which is the duration of the simulation, we also can read a file if one exists. And that file by default here, is named PARINIT. And therefore before we start this calculation, what we need to do is rename that file. That is, if you go to hysplit4/working, you'll see there's a PARDUMP file here. That was the output from the previous simulation, so let's rename that to PARINIT, and so when the model runs, it starts up, it will see this file and it will load the particle positions in that file, which are all valid at 12 UTC. If that particle file had multiple time periods in it, which it could, you could output particle files at multiple intervals. And so a file can contain multiple time periods. HYSPLIT will go through each of the time periods in that file and determine if any one of them matches the starting time of the model simulation, and then it will load those particles for that starting time.

Also, at this point all you need to do is run model.

Now this calculation is going to run a little bit slower, because instead of starting out with zero particles, it starts out with 5700 particles, and then it adds more particles as

the release occurs between hours 12 and 24.

And we of course created a new PARDUMP file and just to confirm, I can actually look at the MESSAGE file here. I don't need to go through the graphical user interface. You can see that if we go down to the startup phase here, there will be a message that says that the particle file was opened, and we've loaded 5700 particles, and as the calculation starts, we are adding to that particle number. So therefore, if we do a display, particle, we will now have 11,400 particles. And you can see this area here represents the initial 12 hour period, these particles are the furthest downstream, but they were released from the ground to six thousand. The new release occurred here and these are all from the ground and 12,000 m.

Now we should take this one more step, that is until the 15th, or starting on the 15th, because this plot was valid on the15th at 00 00. So to go 12 more hours, you should know what to do, we set up the run and we start at 15 00 and let's change the height back to 6, again just that is visible in the particle distribution. And we should rename, now we have a PARINIT already, so let me just get rid of that one here, and just say PARINIT_h12 and the new PARDUMP is now renamed to PARINIT. So the model will read that one, so we run model.

And of course it is running slower again because we started now with 11,400 particles.

So this process that we are doing here is a manual

process. In another section, in this volcanic ash section, we will look at creating a single file, where we can define a complex emission scenario. In this situation, we manually created this scenario with three different release heights.

Let's take a look at it and now you can see, again the complex scenario with the initial section to 6 km, the second 12 hour period to 12 km, and then back again to 6 km. And of course the downwind pattern is quite complex. And as I said we will look at more complex or more automated ways of doing this. This is this is a manual technique.

Now the technique can also be used for not just changing emission characteristics, but it could also be used in a forecasting environment. In that we continually update the model with archive data, analysis data, to generate the initial conditions or the particle file, that we could use to do a forecast or projection. And then after 12 hours we would get the new analysis data or after 6 hours depending upon the frequency, and we could then update the analysis part of the calculation by moving the particle positions forward another 6 or 12 hours. And then from that running another simulation to do a forecast out to 48 or 72 hours. So the use of the particle file applies to many different kinds of scenarios.

And this concludes the section on restarting the model from the PARDUMP file.